CFD for a Research Reactor—A Step Toward Power Reactors

Rizwan-uddin

UNIVERSITY OF ILLINOIS
AT URBANA-CHAMPAIGN
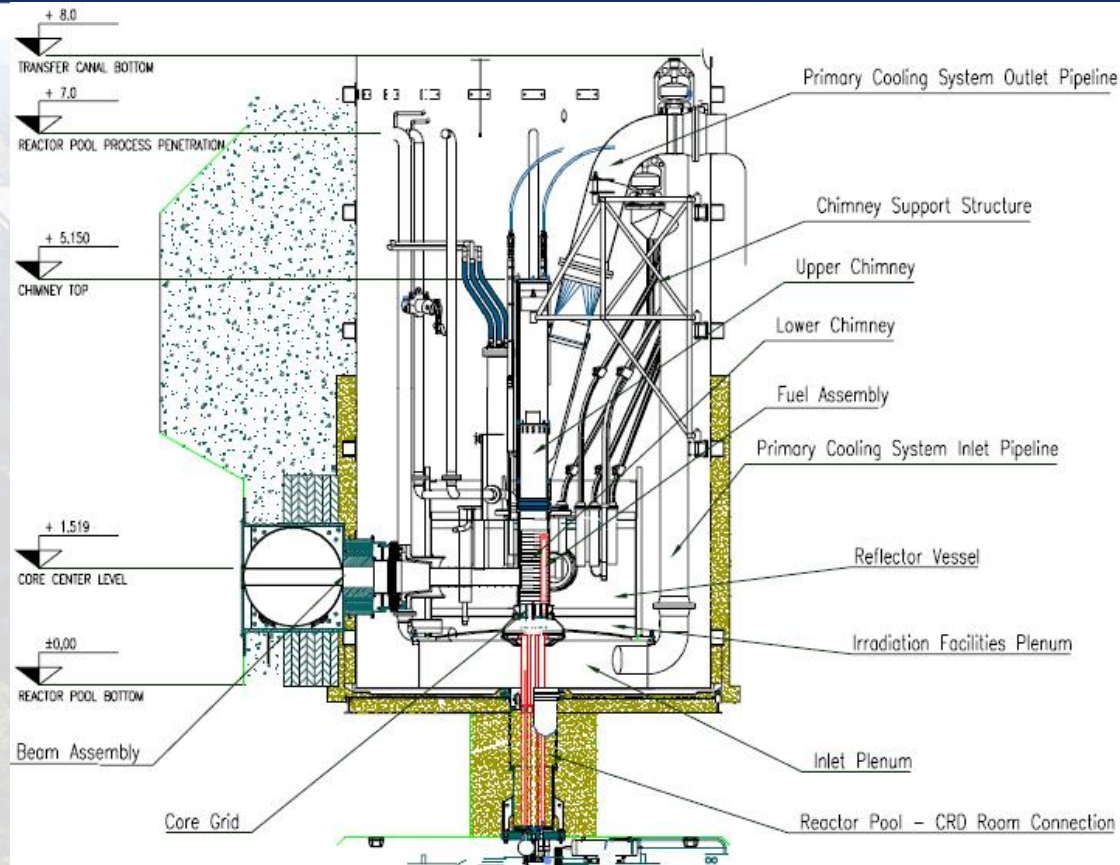
Introduction

- The cooling system of Replacement Research Reactor (RRR)
- CFD simulation of RRR
  - Assembly level simulation
  - Porous media parameters
  - Full Scale Natural Circulation Simulation
- A Power Reactor

1.1 Replacement Research Reactor (RRR)

Diagram labels: Transfer Canal Bottom (+8.0), Reactor Pool Process Penetration (+7.0), Chimney Top (+5.150), Core Center Level (+1.519), Reactor Pool Bottom (±0,00); Beam Assembly; Core Grid; Primary Cooling System Outlet Pipeline; Chimney Support Structure; Upper Chimney; Lower Chimney; Fuel Assembly; Primary Cooling System Inlet Pipeline; Reflector Vessel; Irradiation Facilities Plenum; Inlet Plenum; Reactor Pool – CRD Room Connection

- RRR is a research reactor design has been constructed in Argentina and Australia.
- RRR is multi-purpose open-pool reactor.
- The core is located 13 meters below the pool surface.
- Two different coolant circulation mechanisms are designed for RRR.
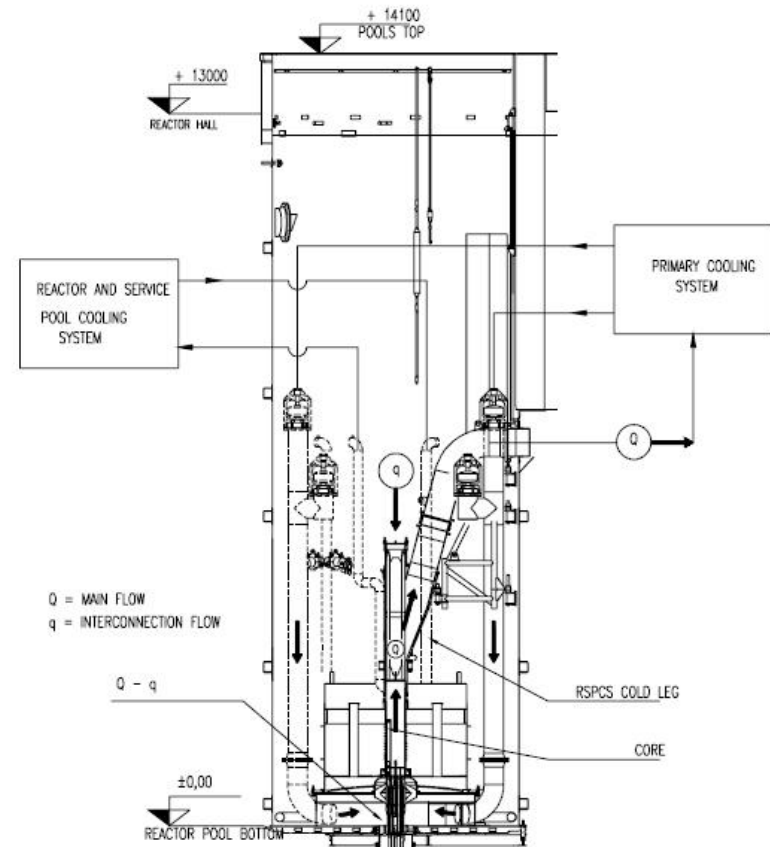- Forced Circulation
- Natural Circulation

1.2 Forced Circulation under Normal Operation

- Forced circulation under normal operating conditions.
- 20MW thermal.
- 90% coolant pump flow rate goes through reactor core for cooling.
- 10% coolant pump flow rate flow downward in chimney to keep the upper part of reactor pool free of radioactive material.
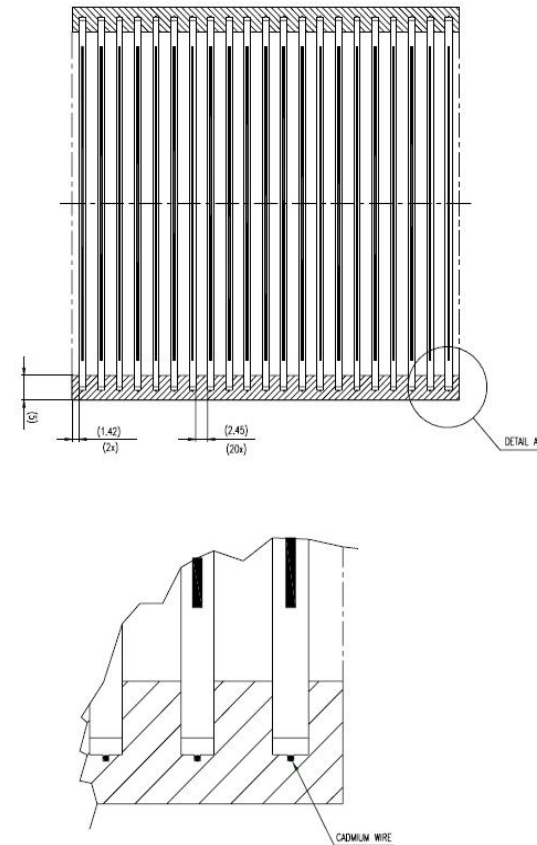
- After reactor shutdown, natural circulation sets up.

- The natural circulation flow in the chimney is upward.
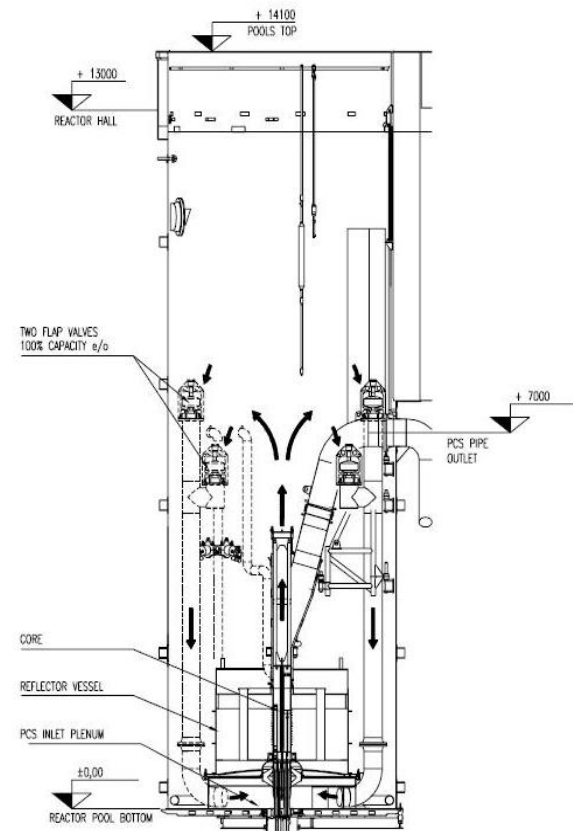
1.4 Fuel Assembly of RRR

- A fuel assembly has 21 fuel plates and 22 coolant channels. The core of RRR has 16 fuel assemblies in the configuration of 4X4 grid.

- There are total of 336 fuel plates (1.5 x 65 x 615mm) and 352 (2.54 x 75 x 1000mm) rectangular flow channels.

2.1 CFD simulation of RRR

- Gambit 2.0 was used to generate the geometry model and mesh.
- A fuel assembly was modeled to study the relation between core pressure drop and flow rate.
- Fuel assembly simulation results were used to determined the parameters in porous media model for full scale RRR simulation.
- The natural circulation of RRR in full scale was simulated.
- Porous media model was applied to simplified the mesh for RRR core.
- Boussinesq Approximation was turned on in the natural circulation simulation.
- Standard k-ε model with enhanced wall treatment was selected for the turbulent flow.

2.2 Assembly CFD Simulation
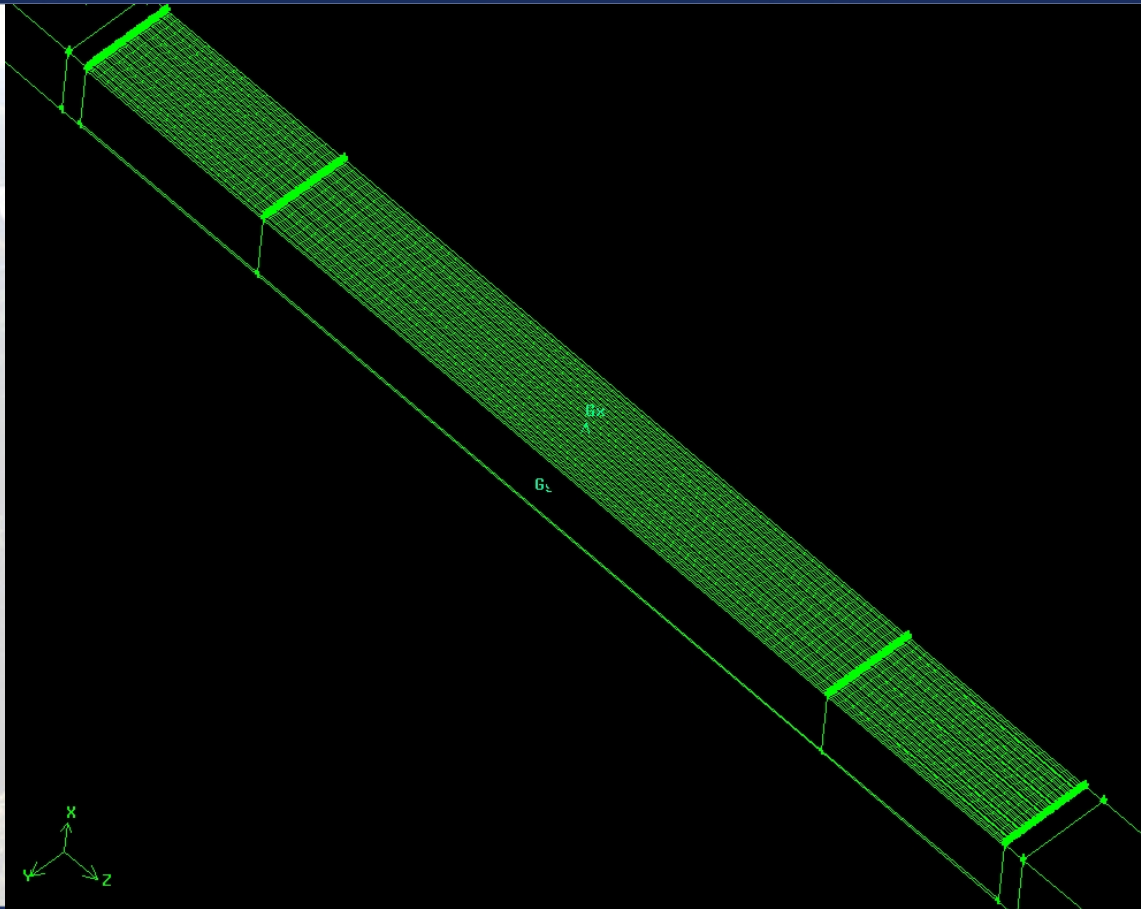
- A RRR fuel assembly is simulated by Fluent 6.2
- Average velocity through the core under normal operating conditions (forced flow) is 8.2 m/s.
- A set of simulations with different inlet velocity (0.1m/s~ 5.08m/s) have been carried out.
- Mesh refinement study has been done with 2.2M cell and 2.9M cell meshes.

2.3 Assembly CFD Simulation

Outlet pressure

B

Fuel plates

A
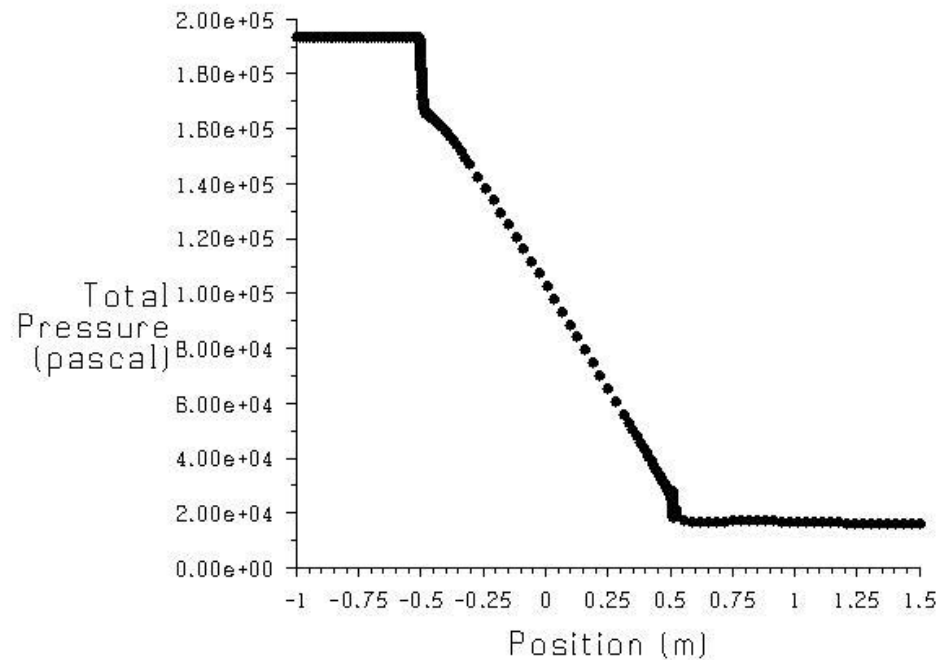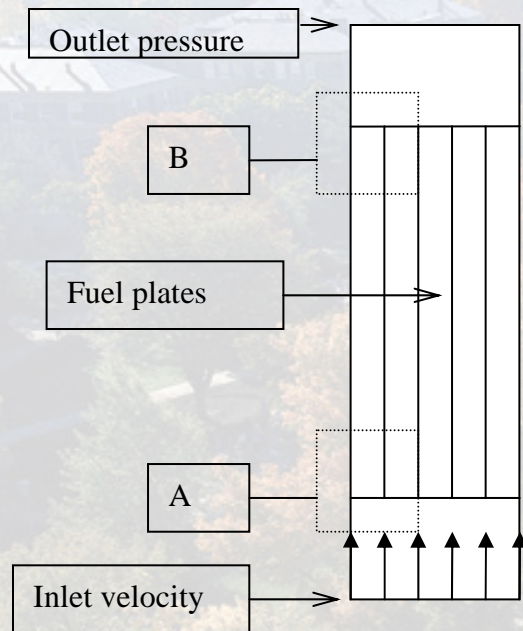
Inlet velocity

2.4 Pressure Drop along the core flow channel

The pressure drops corresponding to different inlet velocity conditions are listed in the Table. The parameters in the porous media model for the natural circulation simulation are determined by curve fitting the pressure drops data D_{ij}, called viscous resistance factor, is 1.33e7 (1/m2) C_{ij}, called inertial resistance factor is, (8.8 1/m)

Table I. Core pressure drop results from assembly level simulations with structural details

Inlet velocity (m/s)	Assembly Pressure Drop with Structural Details (Pa)	Channel Pressure Drop without Fuel Plates (Pa)	Pressure Drop due to the fuel plates (column 2 – column 3) (Pa)
0.01	131.7	0.062	131.6
0.025	349.1	0.20	348.9
0.05	773.3	0.50	772.8
0.1	1650.3	15.03	1635.3
0.5	6188.5	34.39	6154.1
1.0	14819.0	118.12	14700.9
2.0	45279.9	412.80	44867.1
5.08	180980.8	2259.98	178720.8

The additional source term in the momentum equation to account for the porous media zone.

$$S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho v_{mag} v_j \right)$$

2.5 Porous Parameters and Pressure drop comparison

Table II. Core pressure drop results comparison

Inlet velocity	Pressure Drop across the Assembly (with Structural Details)	CFD for Pressure Drop across the Assembly (with Porous Media*)			CFD for Pressure Drop across the Assembly (with Porous Media**)		
(m/s)	(Pa)	C_z (1/m)	D_z (1/m^2)	Δp (Pa)	C_z (1/m)	D_z (1/m^2)	Δp (Pa)
0.01	131.7			135.7			132.2
0.025	349.1			340.1	119.3	1.25E+07	357.7
0.05	773.3			683.0			779.9
0.1	1650.3	8.8	1.59E+07	1388			1422
0.5	6188.5			7844			7983
1.0	14819.0			17953	8.58	1.37E+07	16202
2.0	45279.9			44933			45170
5.08	180980.8			184438			187167

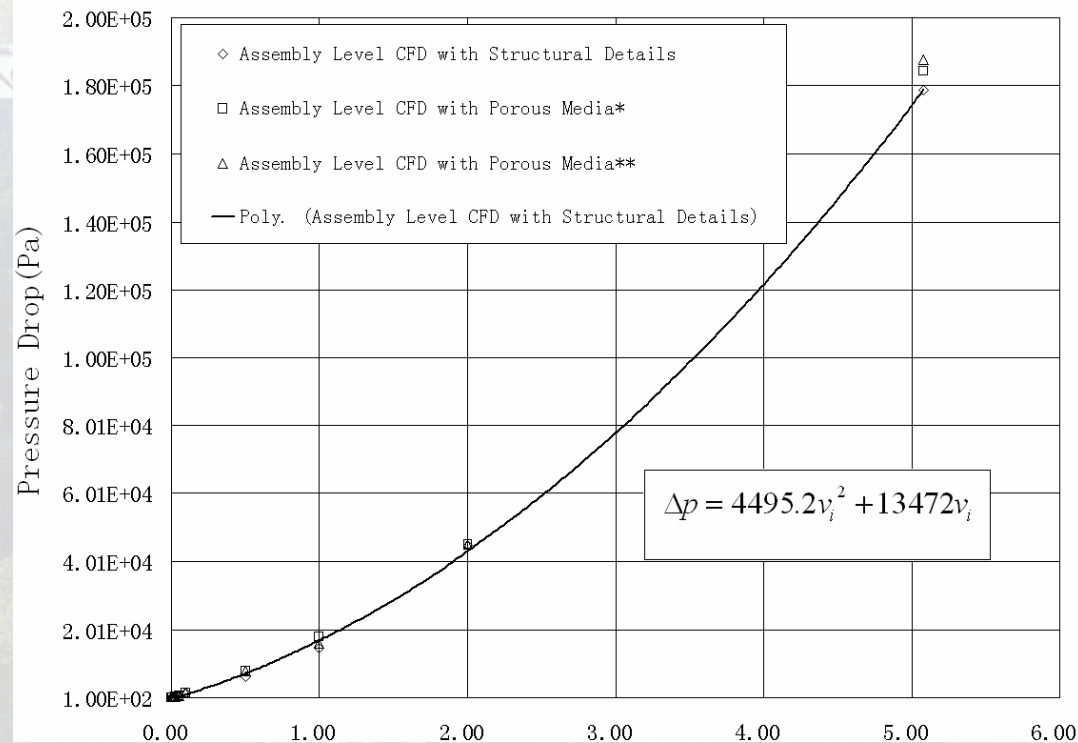* using a single set of values for C_z and D_z for the entire velocity range

** using separate C_z and D_z for low velocity range and high velocity range

2.6 Curve Fitting Pressure Drop to find porous parameters

Pressrue Drop in a Reactor Assembly

◇ Assembly Level CFD with Structural Details

□ Assembly Level CFD with Porous Media*

△ Assembly Level CFD with Porous Media**

— Poly. (Assembly Level CFD with Structural Details)

Pressure Drop (Pa)

2.00E+05
1.80E+05
1.60E+05
1.40E+05
1.20E+05
1.00E+05
8.01E+04
6.01E+04
4.01E+04
2.01E+04
1.00E+02

0.00   1.00   2.00   3.00   4.00   5.00   6.00

$$\Delta p = 4495.2 v_i^{2} + 13472 v_i$$

* using <u>a single set of values for</u> C_z and D_z

for the <u>entire</u> velocity range.

** using separate C_z and D_z

for low velocity range and high velocity range

2.7 Velocity Vector at Zone A

Velocity Vectors Colored By Z Velocity [m/s]   Feb 13, 2005
                                              FLUENT 6.1 (3d, dp, segregated, ske)

2.8 Velocity Vector at Zone B

Velocity Vectors Colored By Z Velocity (m/s)

Feb 13, 2005
FLUENT 6.1 (3d, dp, segregated, ske)
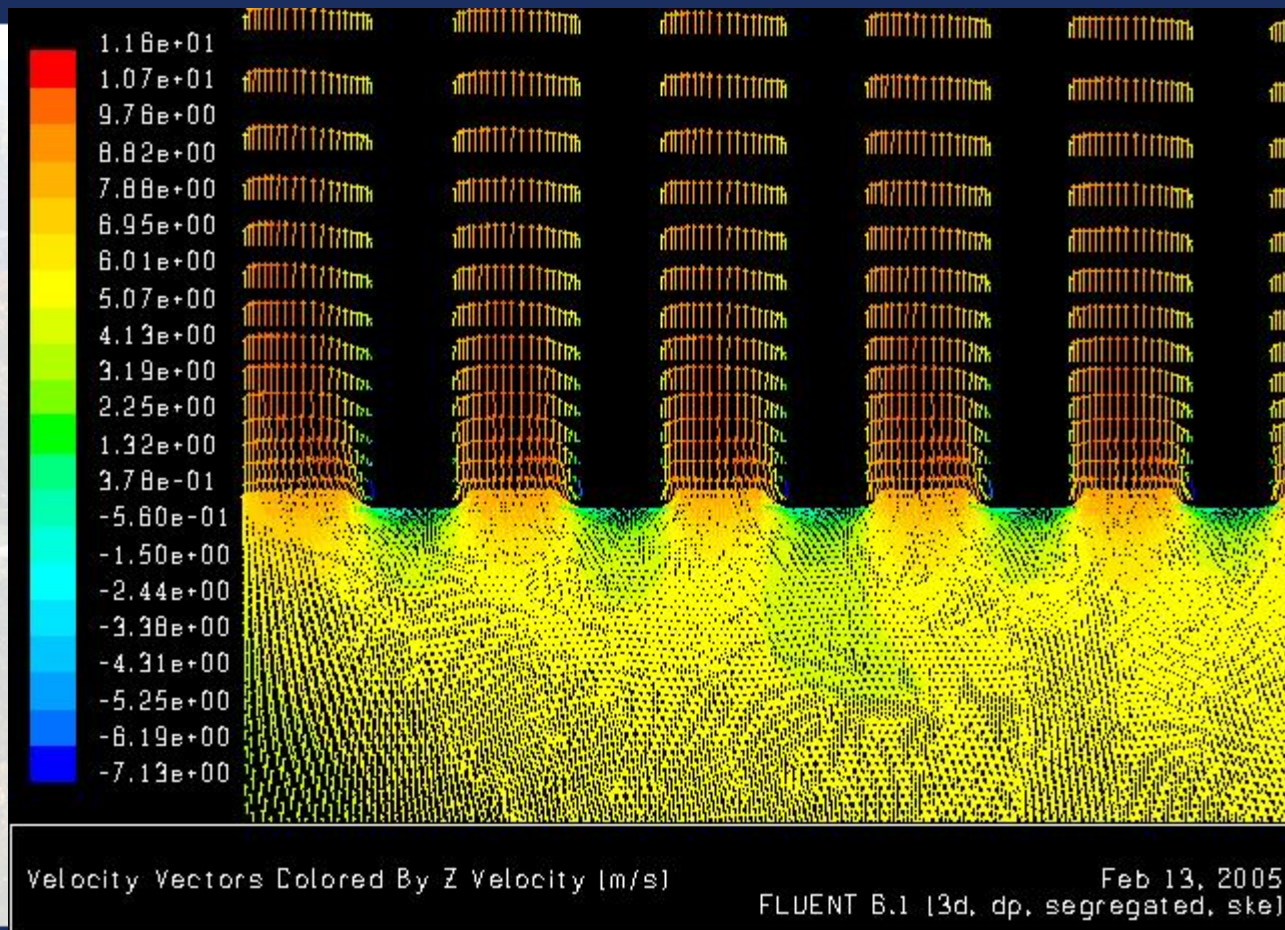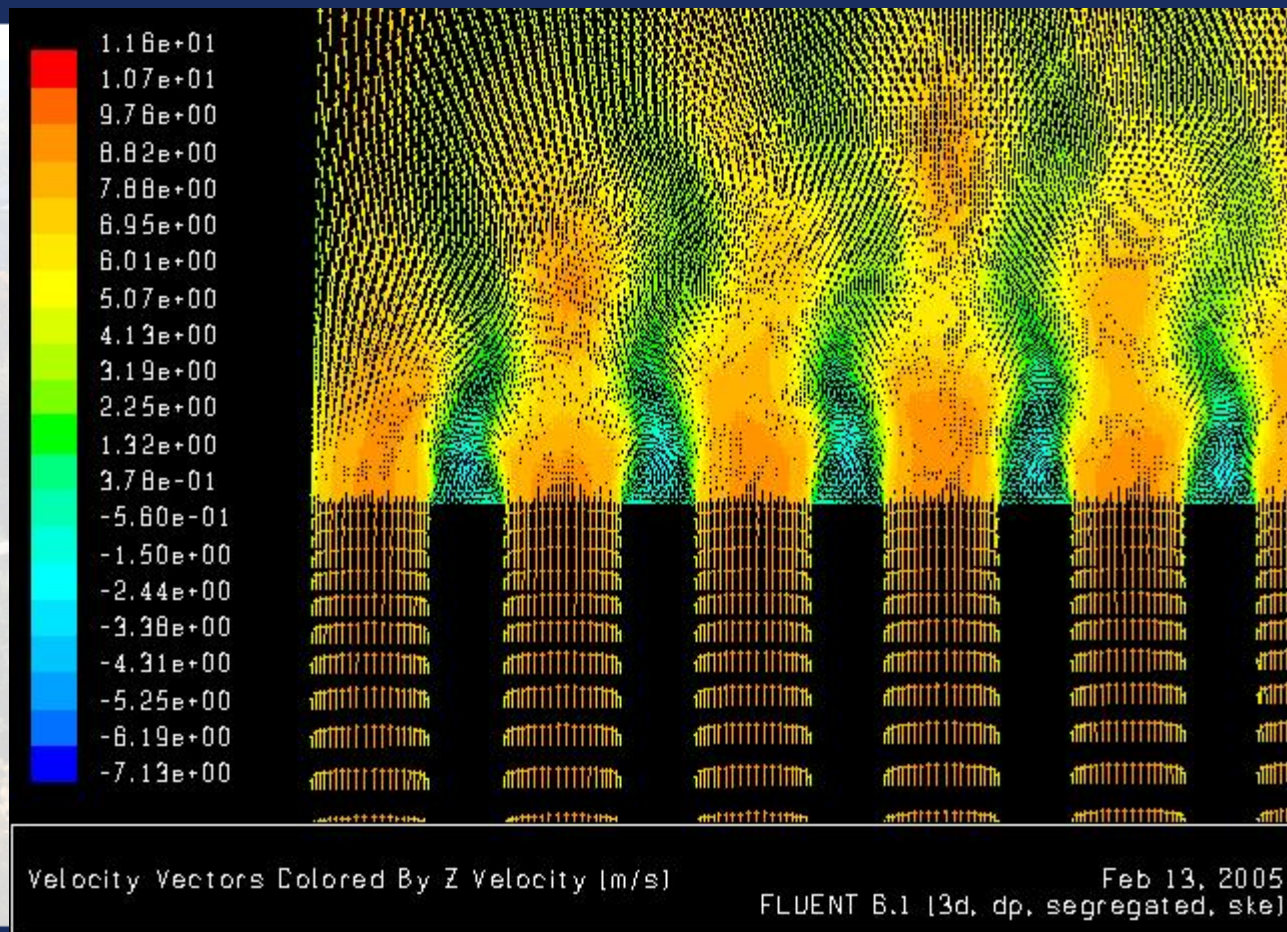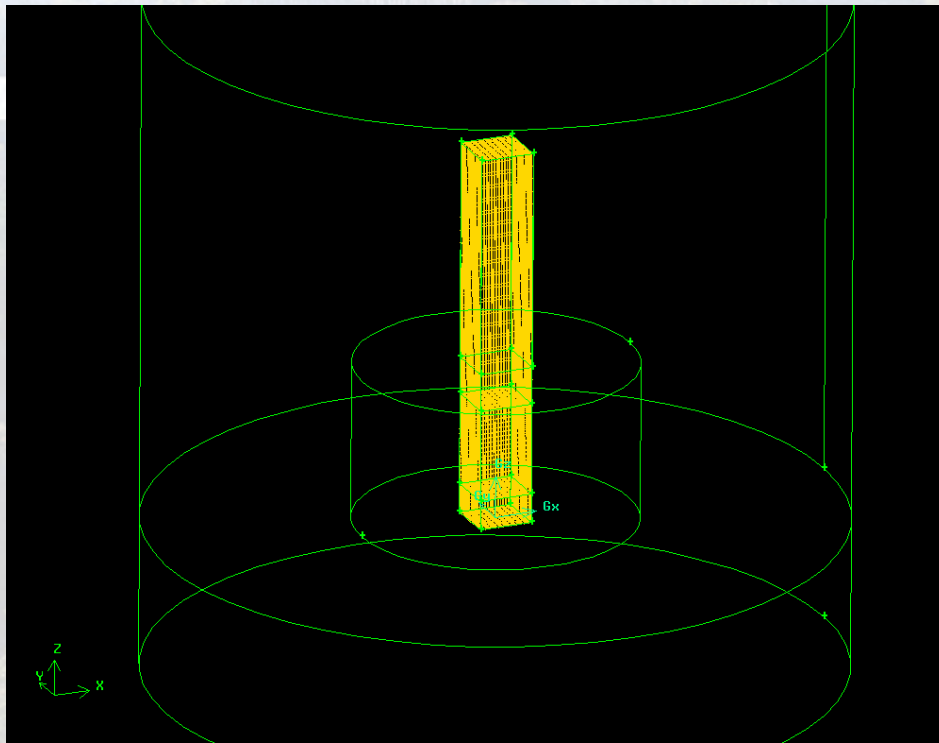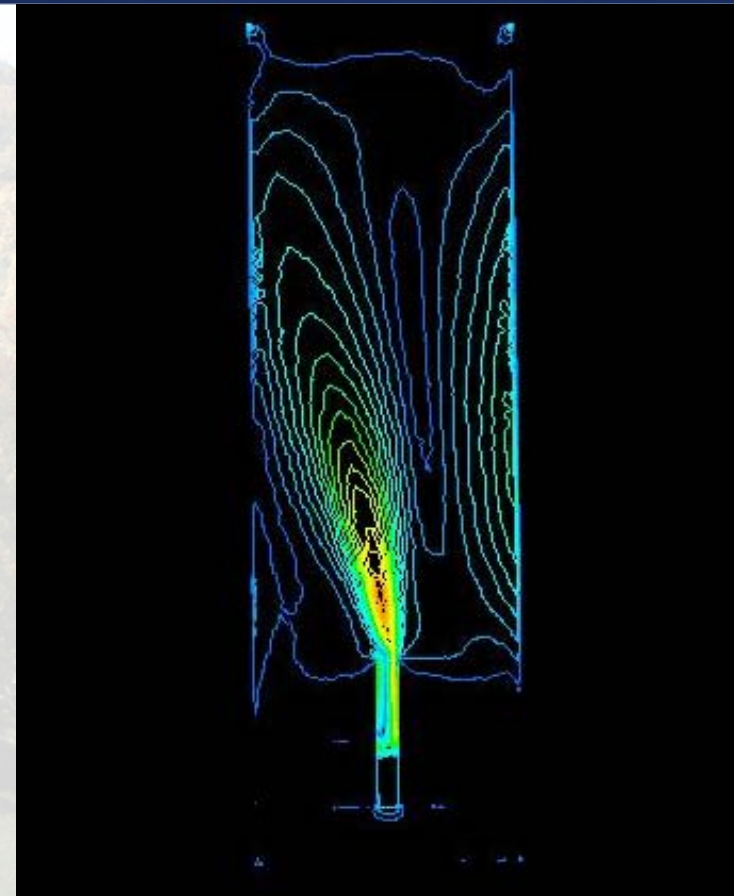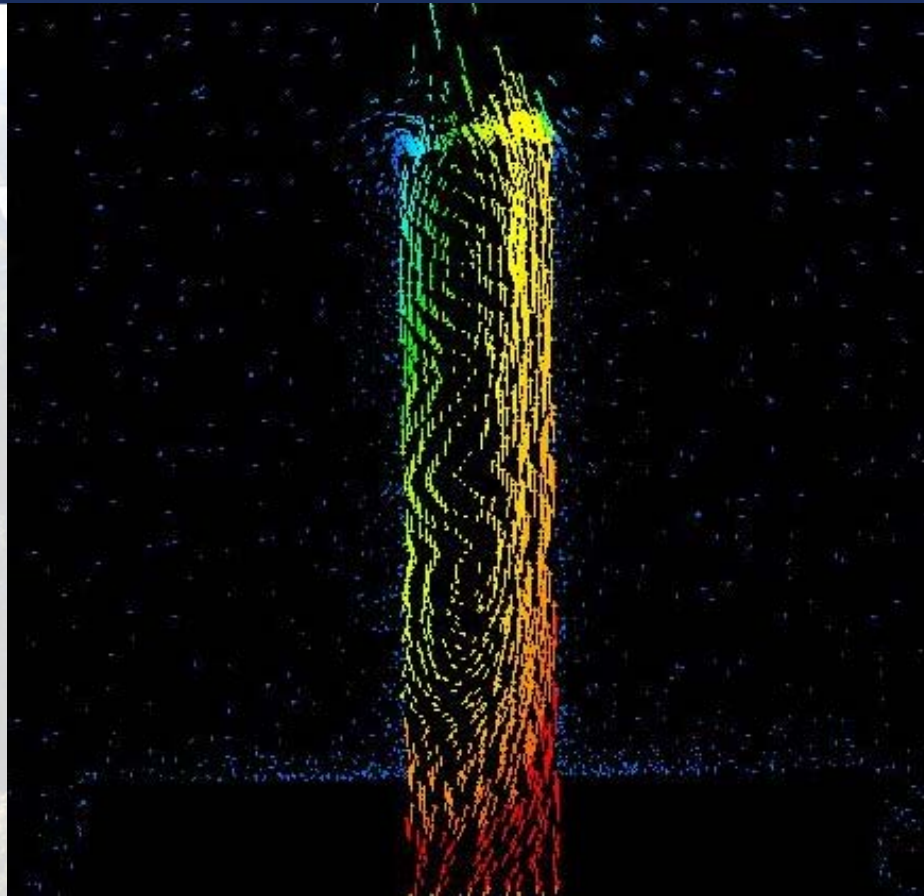
2.9 Model for RRR Natural Circulation
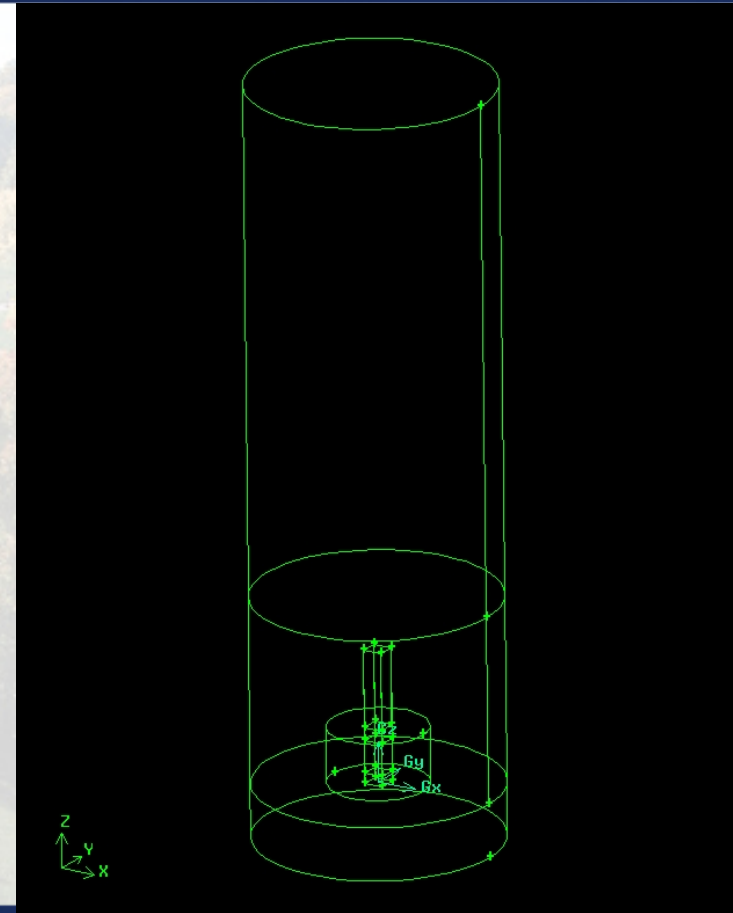
2.10 RRR Natural Circulation Simulation

- The RRR was simplified to a core, a chimney, a heavy water reflector tank and a reactor pool.
- An inner natural circulation was observed inside the chimney.
- The highest natural flow velocity happened above the chimney exit.
- The temperature increase between core inlet to core outlet is 49K.
- The flow rate is 5.11kg/s through the core.

2.11 RRR Natural Circulation Simulation

2.12 CFD Parameter Study for RRR Natural Circulation

- The parameter sensitivity of chimney dimension has been studied. Simulation with chimney of 80% of original width was compared with results with simulation with full size chimney.

- No inner circulation in chimney was found in 80% case.

- The comparison suggests inner circulation in chimney may occur in natural circulation case and is harmful for the heat removal ability.

Table III. Natural circulation conditions for two different chimney sizes

Chimney Cross-sectional Area (m^2)	Coolant flow rate thought core (kg/s)	Temperature rise across core (K)	Average velocity in chimney (m/s)
0.1225	5.11	49	0.0418
0.0784	5.26	46	0.0672

Research to Power

- Simulation of the IRIS (power) reactor

3.2. IRIS CFD Group

- IRIS (International Reactor Innovative and Secure) under development for three years
- An international consortium of over twenty organizations from nine countries.
- UIUC has been participating with the CFD group of IRIS consortium since Jan, 2004.
- The CFD group is:
  - (1) Politecnico di Milano
  - (2) University of Illinois at Urbana-Champaign
  - (3) Tokyo Institute of Technology
  - (4) Università di Pisa
  - (5) Westinghouse, Science & Technology Center
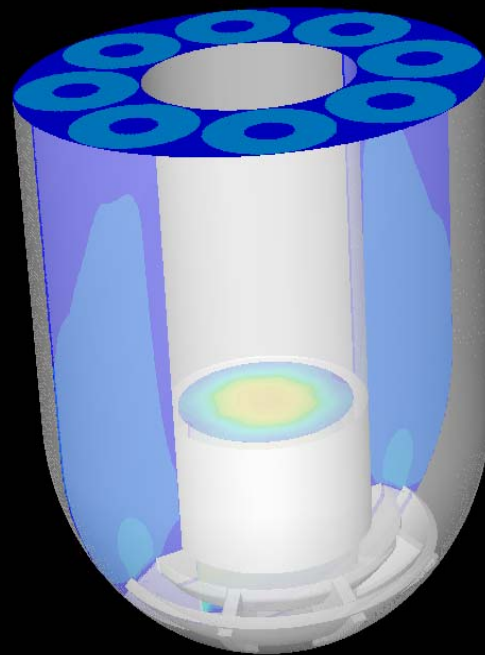
The mesh by
POLIMI

3.2.2. The IRIS CFD Simulation

- IRIS Simulation employs 10M cells for CFD geometry model.

- This simulation challenge the computational ability of Tungsten.

- It takes 8 GB memory to load the problem in, and can't be handled by a single workstation.

- Interactively, problem generation and initiation takes 4 computation nodes of Tungsten

- In batch mode running, the problem was run on 16 computation nodes, which have 32 processors.

3.3. The Downcomer Mixing

- Problem: CFD simulation of flow mixing of different temperatures or different boron concentration in the downcomer and lower plenum.

- The mesh is generated by Gambit 2.0 on Copper cluster. The simulation is carried out by Fluent 6.1 on Copper with 4-16 processors.

- K-Epsilon model has been chosen to model turbulence.

- A set of simulations with different mesh complexities have been used to study the effect of mesh on CFD results.

- In the core region, porous media model and fuel bundle lumped cylinder model results are being obtained and compared (in progress).
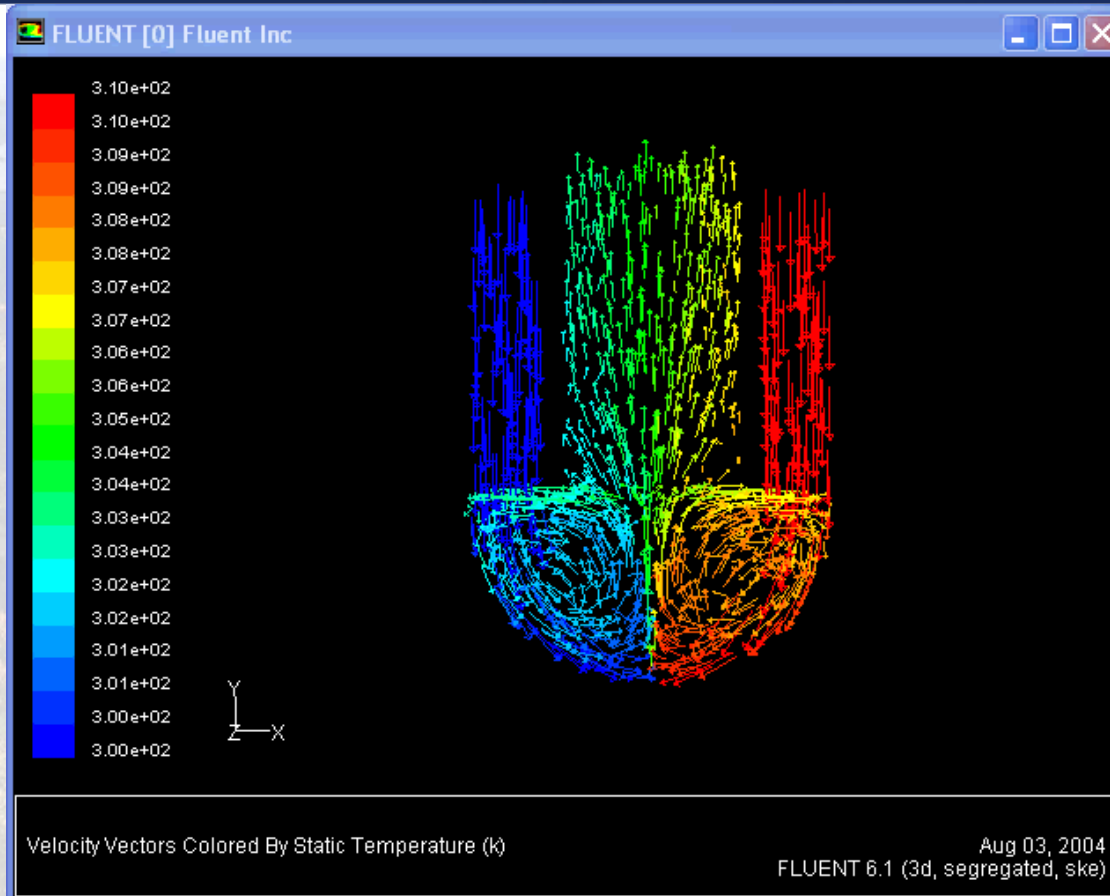
3.3.1 Downcomer CFD Model
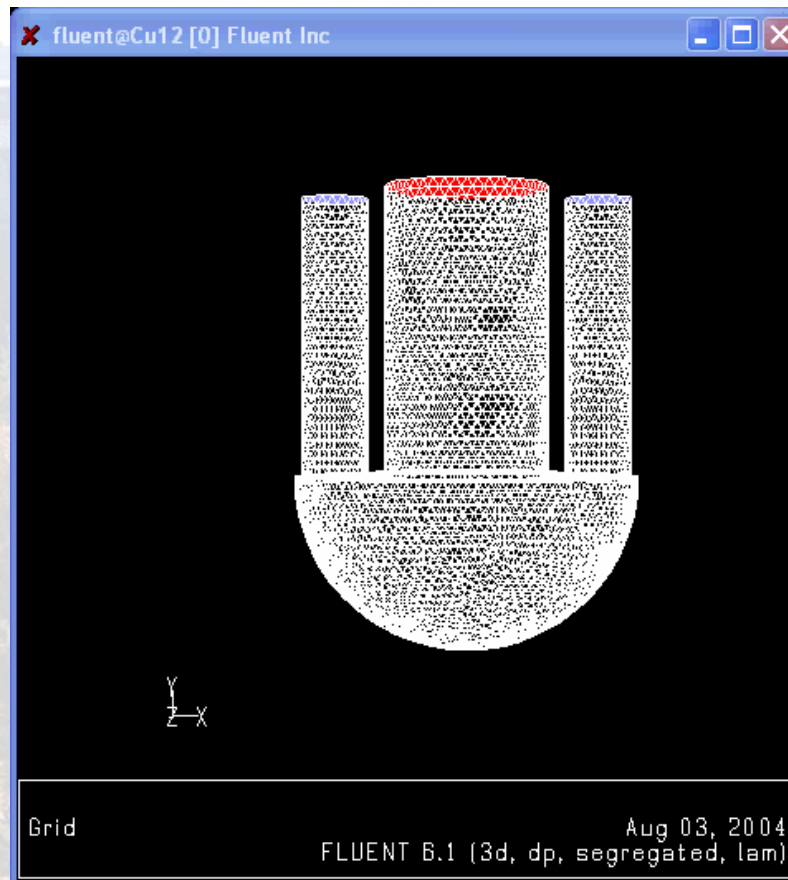
3.3.2. Velocity vector plots in a plane. Color represents temperature (K).

Velocity Vectors Colored By Static Temperature (k)

Aug 03, 2004
FLUENT 6.1 (3d, segregated, ske)

Rocket Center at UIUC (ASCI)

- **Physical Science and Engineering Teams**
- **Combustion and Energetic Materials
  Fluid Dynamics
  Structures and Materials**
- **Computer Science Teams**
- **Computational Environments
  Computational Math and Geometry**
- **Multidisciplinary Teams**
- **Validation and Specification
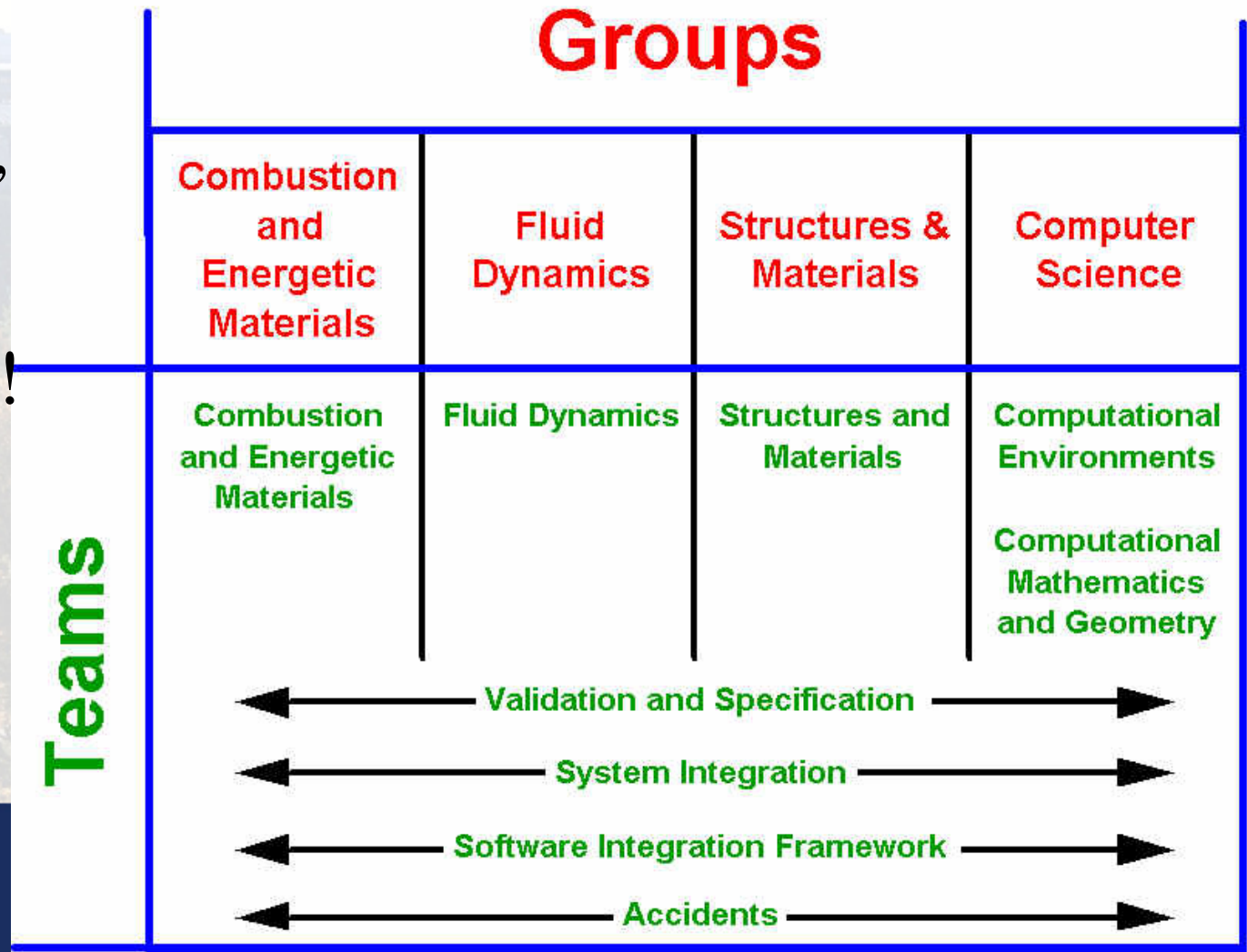 System Integration
  Software Integration Framework
  Accidents**

Rocket Center

Take out "combustion" and insert "neutronics"!!!!

Rocket Center

- Mostly started with legacy codes; but evolved to fairly advanced systems.

- Nuclear can do the same

Suggestion (to start; already under way)

- Couple system codes like RELAP with CFD.
- Gradually decrease components modeled using RELAP and replace by CFD.
- Add detailed neutronics
- etc

3.1 Computational Resources at UIUC

- Computation was carried out on Linux cluster and IBM P690 parallel computer. The number of the processors use in the simulation was up to 8.
- NPRE at UIUC can access the super computation resource as the user of the clusters of NCSA.
- NCSA is The National Center for Supercomputing Applications.
- The Clusters of NCSA at UIUC
  - copper.ncsa.uiuc.edu
  - tungsten.ncsa.uiuc.edu
  - TeraGrid Project
- NCSA has the parallel Fluent license allowing up to use 128 processors.

Summary

- Full research reactor and pool simulations were carried out using the porous media model for the core region.

- Mesh for natural circulation simulation ranges from 400k-2M cells. Mesh for fuel assembly has 2.2 M cells.

- In natural circulation simulation, with a "constant" residual heat of 1 MW (5% of full thermal power) added to reactor core porous media zone as thermal source term, the temperature rise across the core under natural circulation conditions is found to be around 49 K.